RTO-TR-27

NORTH ATLANTIC TREATY ORGANIZATION

RT

RESEARCH AND TECHNOLOGY ORGANIZATION

BP 25, 7 RUE ANCELLE, F-92201 NEUILLY-SUR-SEINE CEDEX, FRANCE

RTO TECHNICAL REPORT 27

A Feasibility Study of Collaborative Multi-facility Windtunnel Testing for CFD Validation

(Etude de faisabilité d'essais en soufflerie multisites menés en coopération, en vue de la validation des calculs de l'aérodynamique numérique - CFD)

Report of the RTO Applied Vehicle Technology Panel (AVT) Working Group 011.
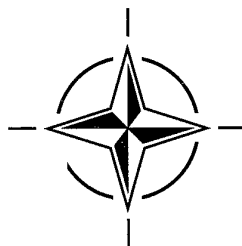
20000518 106

The Research and Technology Organization (RTO) of NATO

RTO is the single focus in NATO for Defence Research and Technology activities. Its mission is to conduct and promote cooperative research and information exchange. The objective is to support the development and effective use of national defence research and technology and to meet the military needs of the Alliance, to maintain a technological lead, and to provide advice to NATO and national decision makers. The RTO performs its mission with the support of an extensive network of national experts. It also ensures effective coordination with other NATO bodies involved in R&T activities.

RTO reports both to the Military Committee of NATO and to the Conference of National Armament Directors. It comprises a Research and Technology Board (RTB) as the highest level of national representation and the Research and Technology Agency (RTA), a dedicated staff with its headquarters in Neuilly, near Paris, France. In order to facilitate contacts with the military users and other NATO activities, a small part of the RTA staff is located in NATO Headquarters in Brussels. The Brussels staff also coordinates RTO's cooperation with nations in Middle and Eastern Europe, to which RTO attaches particular importance especially as working together in the field of research is one of the more promising areas of initial cooperation.

The total spectrum of R&T activities is covered by 7 Panels, dealing with:

- SAS   Studies, Analysis and Simulation
- SCI   Systems Concepts and Integration
- SET   Sensors and Electronics Technology
- IST   Information Systems Technology
- AVT   Applied Vehicle Technology
- HFM   Human Factors and Medicine
- MSG   Modelling and Simulation

These Panels are made up of national representatives as well as generally recognised 'world class' scientists. The Panels also provide a communication link to military users and other NATO bodies. RTO's scientific and technological work is carried out by Technical Teams, created for specific activities and with a specific duration. Such Technical Teams can organise workshops, symposia, field trials, lecture series and training courses. An important function of these Technical Teams is to ensure the continuity of the expert networks.

RTO builds upon earlier cooperation in defence research and technology as set-up under the Advisory Group for Aerospace Research and Development (AGARD) and the Defence Research Group (DRG). AGARD and the DRG share common roots in that they were both established at the initiative of Dr Theodore von Kármán, a leading aerospace scientist, who early on recognised the importance of scientific support for the Allied Armed Forces. RTO is capitalising on these common roots in order to provide the Alliance and the NATO nations with a strong scientific and technological basis that will guarantee a solid base for the future.

A Feasibility Study of Collaborative Multi-facility Windtunnel Testing for CFD* Validation

(RTO TR-27)

Executive Summary

The Applied Vehicle Panel of the RTA established in 1997 a Working Group to study the feasibility of collaborative, multi-facility windtunnel testing for CFD Validation. CFD validation is generally appreciated as a very important part of the development of Computational Fluid Dynamics. Dedicated, well controlled and accurate windtunnel tests are considered as the main source of CFD validation. Windtunnels, however, all have their own specific bias errors associated with practical constraints (like tunnel wall and model support systems) and flow imperfections. Testing of one model in several facilities can better establish the level of uncertainty involved in windtunnel testing. This is important too for the assessment of possible differences between CFD and the windtunnel test.

At the start of the Working Group a limited inquiry was made (using a questionnaire that was distributed within the CFD community) to probe the experiences with CFD validation. Half of the respondents found the situation acceptable, with satisfactory and disappointing experiences equally divided on both sides. Most of the respondents indicated a need for more detailed measurements. A lack of validation experiments for helicopters was noted.

The Working Group assessed the need for CFD validation for three different configurations: fighters, transport aircraft and missiles. In the course of the study, helicopters were added. For all these configurations there is a qualified need for additional CFD validation experiments:

- **fighter aircraft:** only useful for a representative 'stealth-like' configuration; this would involve the design and manufacturing of a new generic windtunnel model; interest focuses on mixed attached/separated flows with vortex flow involving complex measuring techniques;

- **transport type aircraft:** there is an interest in low speed / high lift configurations, possibly using already existing windtunnel models like the DLR 'ALVAST' or the B737-100;

- **missiles:** there is a need for validating complex configurations although the multi-facility aspect is less pronounced;

- **helicopters:** there is a clear lack of detailed experiments in view of the flow field complexity; experience in measuring techniques could be combined through multi-facility testing experience.

Interest is focussed on validating complex flow phenomena, involving interactions of different flow types. It is essential to apply new, challenging measuring techniques to obtain flow field information that, up till now, has been very limited. Also, in validation experiments, measuring uncertainties can be reduced by careful design of the experiment. Therefore, the technical need for and cost effectiveness of multi-facility testing to reduce possible bias effects in relation with CFD validation, is limited. This doesn't mean that there is no need for collaborative multi-facility windtunnel testing:

- From the point of view of windtunnel test techniques it is still of interest to establish bias errors between various facilities using comparative tests with one or more simple standard models, specifically designed and instrumented to understand the cause of possible differences. Such an exercise should be viewed in the context of 'quality assurance'.

- For CFD validation multi-facility testing is of interest to widen the scope and the technical depth of the experiment. It will be cost effective to use the experience of different groups of experts for advanced measuring techniques in a complementary manner.

For all these cases the Working Group members have made a 'bottom-up' inquiry within their own organisations to find out if one or more of these programs could find support. In all cases the response was inconclusive. Cost issues and commercial / political reasons were quoted to limit the chances for a multi-facility exercise. It is therefore believed that such a program could only be envisaged when there is a clear top-down commitment from the organisations involved.

---

*CFD: Computational Fluid Dynamics.

Etude de faisabilité d'essais en soufflerie multisites menés en coopération, en vue de la validation des calculs de l'aérodynamique numérique - CFD

(RTO TR-27)

Synthèse

En 1997, la commission sur la technologie appliquée aux véhicules (AVT) de la RTA, a créé un groupe de travail pour étudier la faisabilité d'essais en soufflerie multisites menés en coopération en vue de la validation des calculs de l'aérodynamique numérique (CFD). La validation est généralement considérée comme une étape très importante du développement de l'aérodynamique numérique. Des essais en soufflerie spécialisés et rigoureux, effectués dans des conditions bien maîtrisées sont la principale source de validation du CFD. Cependant, chaque soufflerie a ses propres biais associés à des contraintes pratiques spécifiques (comme par exemple les effets de paroi et les supports de maquette), et des imperfections d'écoulement. Tester la même maquette dans différentes installations permet de mieux établir le degré d'incertitude lié aux essais en soufflerie. Cela est également important pour l'évaluation d'éventuelles différences entre les calculs CFD et les résultats des essais en soufflerie.

Au début des travaux du groupe de travail, un sondage a été effectué (à l'aide d'un questionnaire diffusé aux spécialistes du CFD) afin de recueillir l'expérience en matière de validation CFD. La moitié des personnes interrogées trouvait la situation acceptable, avec autant d'expériences satisfaisantes que de résultats décevants. La plupart d'entre elles ont évoqué le besoin de mesures plus pointues. L'absence d'essais de validation pour les hélicoptères a aussi été noté.

Le groupe de travail a relevé un besoin de validation CFD pour trois catégories différentes : les avions de combat, les avions de transport et les missiles. Les hélicoptères ont été ajoutés à cette liste au cours du projet. Pour toutes ces catégories il existe un besoin précis d'expériences de validation CFD.

- **Les avions de combat :** intéressant uniquement pour une configuration représentative du type « furtif », impliquant la conception et la réalisation d'une nouvelle maquette générique pour essais en soufflerie ; les domaines d'intérêt sont les écoulements mixtes attachés/décollés avec écoulements tourbillonnaires faisant appel à des techniques de mesure complexes ;

- **Les avions de transport :** il est intéressant d'étudier des configurations faible vitesse/portance élevée, en utilisant, éventuellement des maquettes de soufflerie existantes, comme l'ALVAST du DLR ou le B737-100 ;

- **Les missiles :** il y a lieu de valider des configurations complexes, bien que l'aspect multisites soit moins prononcé ;

- **Les hélicoptères :** vu la complexité du champ d'écoulement, il y a une insuffisance manifeste d'expérimentation détaillée ; l'expérience acquise en matière de techniques de mesure pourrait être mise en commun par la réalisation d'essais sur des sites multiples.

Le centre d'intérêt commun est la validation de phénomènes d'écoulement complexes, mettant en jeu des interactions entre différents types d'écoulements. Il est essentiel d'appliquer de nouvelles techniques de mesure ambitieuses pour obtenir les données sur le champ d'écoulement qui ont manqué jusqu'à présent. De même, dans les expériences de validation, les incertitudes sur les mesures pourraient être réduites par une conception plus rigoureuse de l'expérience. De plus, le besoin technique et la rentabilité des essais multisites réalisés dans l'optique de réduire d'éventuels biais dans la validation CFD, demeurent limités. Pourtant, cela ne veut pas dire que les essais en soufflerie multisites menés en coopération ne présentent pas d'intérêt :

- Du point de vue des techniques des essais en soufflerie, il demeure intéressant de définir les biais entre différentes installations en réalisant des essais comparatifs sur une ou deux maquettes standards simples, spécifiquement conçues et instrumentées, afin de comprendre les raisons d'éventuelles différences. Un tel exercice doit être vu dans le contexte de « l'assurance qualité ».

- Les essais multisites sont intéressants pour la validation CFD, puisqu'ils permettent d'augmenter la portée et la complexité technique de l'expérience. Il serait rentable de profiter de l'expérience mise en commun de différents groupes de spécialistes en matière de techniques de mesure avancées.

Pour tous ces cas de figure, les membres du groupe de travail ont adopté une démarche ascendante au sein de leurs propres organisations afin d'apprécier l'intérêt qui pourrait s'exprimer pour un ou plusieurs de ces programmes. Les réponses n'ont pas été concluantes. Des considérations de coûts et des arguments politico-commerciaux ont été mentionnés comme facteurs limitant les possibilités de campagnes multisites. Il semblerait donc qu'un tel programme ne peut être envisagé qu'en cas d'engagement réel de la part des directions des organisations impliquées.

Contents

Recent Fluid Dynamics Publications
of the RTO Applied Vehicle Technology Panel
and the Former AGARD Fluid Dynamics Panel

AGARDOGRAPHS

Turbulent Boundary Layers in Subsonic and Supersonic Flow
AGARD AG-335, July 1996

Computational Aerodynamics Based on the Euler Equations
AGARD AG-325, September 1994

Scale Effects on Aircraft and Weapon Aerodynamics
AGARD AG-323 (E), July 1994

Design and Testing of High-Performance Parachutes
AGARD AG-319, November 1991

Experimental Techniques in the Field of Low Density Aerodynamics
AGARD AG-318 (E), April 1991

Techniques Expérimentales Liées à l'Aérodynamique à Basse Densité
AGARD AG-318 (FR), April 1990

A Survey of Measurements and Measuring Techniques in Rapidly Distorted Compressible Turbulent Boundary Layers
AGARD AG-315, May 1989

CONFERENCE AND MEETING PROCEEDINGS

Fluid Dynamics Problems of Vehicles Operating Near or in the Air-Sea Interface
MP-15, February 1999

Missile Aerodynamics
RTO Report MP-5, November 1998

Advanced Aerodynamic Measurement Technology
AGARD CP-601, May 1998

Aerodynamics of Wind Tunnel Circuits and Their Components
AGARD CP-585, June 1997

The Characterization & Modification of Wakes from Lifting Vehicles in Fluids
AGARD CP-584, November 1996

Progress and Challenges in CFD Methods and Algorithms
AGARD CP-578, April 1996

Aerodynamics of Store Integration and Separation
AGARD CP-570, February 1996

Aerodynamics and Aeroacoustics of Rotorcraft
AGARD CP-552, August 1995

Application of Direct and Large Eddy Simulation to Transition and Turbulence
AGARD CP-551, December 1994

Wall Interference, Support Interference, and Flow Field Measurements
AGARD CP-535, July 1994

Computational and Experimental Assessment of Jets in Cross Flow
AGARD CP-534, November 1993

High-Lift System Aerodynamics
AGARD CP-515, September 1993

Theoretical and Experimental Methods in Hypersonic Flows
AGARD CP-514, April 1993

Aerodynamic Engine/Airframe Integration for High Performance Aircraft and Missiles
AGARD CP-498, September 1992

Effects of Adverse Weather on Aerodynamics
AGARD CP-496, December 1991

Manoeuvring Aerodynamics
AGARD CP-497, November 1991

Vortex Flow Aerodynamics
AGARD CP-494, July 1991

Missile Aerodynamics
AGARD CP-493, October 1990

EDUCATIONAL NOTES

Fluid Dynamic Research on Supersonic Aircraft
RTO EN-4, November 1998

High-Order Methods for Computational Physics, Published jointly with Springer-Verlag, Germany
RTO EN-5, 1999

REPORTS

High Speed Body Motion in Water
AGARD R-827, February 1998

Turbulence in Compressible Flows
AGARD R-819, Special Course Notes, June 1997

Advances in Cryogenic Wind Tunnel Technology
AGARD R-812, Special Course Notes, January 1997

Aerothermodynamics and Propulsion Integration for Hypersonic Vehicles
AGARD R-813, Special Course Notes, October 1996

Parallel Computing in CFD
AGARD R-807, Special Course Notes, October 1995

Optimum Design Methods for Aerodynamics
AGARD R-803, Special Course Notes, November 1994

Missile Aerodynamics
AGARD R-804, Special Course Notes, May 1994

Progress in Transition Modelling
AGARD R-793, Special Course Notes, April 1994

Shock-Wave/Boundary-Layer Interactions in Supersonic and Hypersonic Flows
AGARD R-792, Special Course Notes, August 1993

Unstructured Grid Methods for Advection Dominated Flows
AGARD R-787, Special Course Notes, May 1992

Skin Friction Drag Reduction
AGARD R-786, Special Course Notes, March 1992

Engineering Methods in Aerodynamic Analysis and Design of Aircraft
AGARD R-783, Special Course Notes, January 1992

ADVISORY REPORTS

A Selection of Test Cases for the Validation of Large-Eddy Simulations of Turbulent Flows
AGARD AR-345, April 1998

Ice Accretion Simulation
AGARD AR-344, Report of WG-20, December 1997

Sonic Nozzles for Mass Flow Measurement and Reference Nozzles for Thrust Verification
AGARD AR-321, Report of WG-19, June 1997

Cooperative Programme on Dynamic Wind Tunnel Experiments for Manoeuvring Aircraft
AGARD AR-305, Report of WG-16, October 1996

Hypersonic Experimental and Computational Capability, Improvement and Validation
AGARD AR-319, Vol. I, Report of WG-18, May 1996

Aerodynamics of 3-D Aircraft Afterbodies
AGARD AR-318, Report of WG-17, September 1995

A Selection of Experimental Test Cases for the Validation of CFD Codes
AGARD AR-303, Vols. I and II, Report of WG-14, August 1994

Quality Assessment for Wind Tunnel Testing
AGARD AR-304, Report of WG-15, July 1994

Air Intakes of High Speed Vehicles
AGARD AR-270, Report of WG-13, September 1991

Appraisal of the Suitability of Turbulence Models in Flow Calculations
AGARD AR-291, Technical Status Review, July 1991

Rotary-Balance Testing for Aircraft Dynamics
AGARD AR-265, Report of WG11, December 1990

Calculation of 3D Separated Turbulent Flows in Boundary Layer Limit
AGARD AR-255, Report of WG10, May 1990

Members of Applied Vehicle Technology Panel Working Group 011 on "A Feasibility Study of Collaborative Multi-facility Windtunnel Testing for CFD Validation"

CANADA -
Dr. M. KHALID
Institute for Aerospace Research
National Research Council
High Speed Aerodyn. Lab – U66
Montreal Road
Ottawa, Ontario K1A 0R6

FRANCE -
Mr. V. SCHMITT
ONERA
B.P. 72
92322 Châtillon Cedex

GERMANY -
Dr. G. REDEKER
Institut für Entwurfsaerodynamik DLR
Postfach 3267
38022 BRAUNSCHWEIG

ITALY -
Dr. A. AMENDOLA
CIRA
Via Maiorise
81043 CAPUA (CE)

NETHERLANDS -
Ir B. ELSENAAR **(Chairman)**
National Aerospace Laboratory NLR
Anthony Fokkerweg, 2
1059 CM AMSTERDAM

UNITED KINGDOM -
Dr. J.L. FULKER
Defence Evaluation & Research Agency
High Speed and Weapons
Aerodynamics Department
BEDFORD MK41 6AE

UNITED STATES -
Dr. J.H. FOX
Sverdcup Inc.
740 Fourth Str.
ARNOLD AFB, TN 37389

Dr. G. SEIBERT
USAF-WL/FIMC, BLDG 450
2645 Fifth Str. Ste. 7
WRIGHT PATTERSON AFB
OH 45433-7913

1   INTRODUCTION

In 1997 the Applied Vehicle Panel (AVP) of the Research and Technology Agency (RTA) of NATO
established the Working Group WG-011 to "study the feasibility of collaborative, multi-facility
windtunnel testing for CFD Validation". Prof. J.W. Slooff and Dr. R. Bengelink, at the time of the
initiative both members of AGARD's Fluid Dynamics Panel (FDP), initiated this study. In the original
proposal the following justification was given:

*"CFD validation is generally appreciated as a very important part of the development, into maturity,
of Computational Fluid Dynamics. Dedicated, well-controlled and accurate windtunnel tests are
considered as the main source of CFD validation. Windtunnels, however, all have their own specific
bias errors associated with flow quality, noise levels, transition tripping, wall interference, support
interference and the like. Moreover, the magnitude of such effects is, in general, not accurately known.
In this situation testing of one model in several facilities is highly desirable in order to better establish
the level of uncertainty involved in windtunnel testing. This is of particular importance for CFD
validation tests."*

The Working Group assessed the need for CFD validation for three different configurations: fighters,
transport and missiles. Also an inquiry was made (using a questionnaire that was distributed within the
CFD community) with respect to the experiences with CFD validation. As a result from this inquiry,
helicopter configurations were added to this study. For each of these configuration types the added
value and practical possibilities for collaborative multi-facility testing were discussed. Each of the
Working Group members used its own contact within the aerodynamic community to find-out if the
views of the WG members were generally supported. This report summarises the findings of the
Working Group.

The Working Group started its activities fall 1997 in Seattle, followed by a meeting spring 1998 in
Sorrento and a final meeting fall 1998 in Amsterdam.

2 WINDTUNNELS AND CFD VALIDATION

2.1 CFD validation and multi-facility testing

by A. Elsenaar (NLR)

2.1.1 Introduction

Windtunnels play an essential rôle in aircraft design. However, due to the rapid development of computers and numerical methods, CFD has matured to such an extent that the designer is now able to apply CFD in the design process. With the advance of CFD, windtunnels are also used to guide and validate the development of improved CFD methods. The requirements for "development testing" and "CFD validation" are not identical[1,2,3].

CFD simulates the flow numerically in space and time co-ordinates. At the beginning of the development of a particular CFD code the flow physics and the related numerics are all important. In its final stage the performance of the code for practical applications has to be assessed.

To validate a CFD code its results are compared with well-controlled experiments obtained in the windtunnel. However, the windtunnel has its own deficiencies (like wall and support effects) and the validation issue can be narrowed down to the question 'how accurately does the CFD code and the windtunnel represent the reality' and, as a follow-up on this 'how can CFD and the windtunnel be used in a complementary way to improve the validity of both methods'.

2.1.2 Characteristics and requirements for CFD validation

In the present context we will restrict ourselves to the flow physics aspects of CFD validation, leaving aside questions of numerical accuracy and geometrical representation.
The first issue to be resolved is the question if the basic flow physics is adequately modelled. So-called 'building block experiments' are of great help here: they address one physical aspect at the time at a sufficiently large scale to allow detailed (flowfield) measurements for a (most often) relatively simple geometry. Table 2.1-1 provides a rather comprehensive list (compiled by *J.H. Fox*) of 'building block experiments' in order of increasing physical complexity.

In the final stages of the validation process one wants to know how well a CFD code is able to predict aerodynamic data (e.g. lift, drag, pitching moment, loads . . .), both with a particular applications in mind (fighters, transport type, missiles, helicopters) and with respect to complex flow physics involved (e.g. mixtures of attached, separated or partly separated flow, flows with shock waves and vortices, flows with unsteady phenomena . . .). Comparisons are made with results from windtunnel experiments, normally for some characteristic flow conditions. For each of these conditions an overall comparison is made in terms of forces and moments and pressure distributions, followed by an in-depth analysis of details of the flow field like boundary layer development, skin friction magnitude and direction, wake or other flow field velocity measurements. Detailed measurements are required to assess if the relevant flow physics is well represented and/or to explain and understand differences between CFD and experiment in the overall characteristics. However, such detailed flow investigations usually require a special experimental set-up involving (large) models such that these specialised measurements can be made (e.g. LDV, PIV, internal and external probes).

If differences are found they might also be contributed to deficiencies in the experimental test results. In the design of the experiment adequate measures are to be taken to assure that there is sufficient information to assess the quality of the experimental results. In another AGARD report[4] recommendations have been made with respect to wind tunnel tests for CFD validation. Some of these recommendations are repeated here:
- establish flow sensitivities to make sure that the CFD comparison is made at a condition that is not very sensitive to small variations in flow conditions;

- if flow sensitivities can be expected, the CFD comparison should be done for a range of test conditions;
- resolve a possible uncertainty in reference flow conditions from measured boundary conditions, either
  - by deriving the wall induced interference flow fields from the measured boundary conditions (e.g. pressures or velocities near the tunnel walls; this is recommended for perforated or slotted wall tunnels);

or
  - by making the CFD comparison with the tunnel walls represented (this is recommended for solid tunnel walls);
- use as much as possible complementary and /or redundant measuring techniques to assess possible probe interference effects and instrument bias errors;
- minimise as much as possible differences between the calculated and the tested configuration (e.g. by measuring model deformations, measuring the transition location, using simple geometry's (e.g. faired inlets), representing the model support in the calculations . . .).

These recommendations still stand today. In addition to this one can also consider to perform the validation experiment in different facilities. Such a multi-facility experiment will greatly improve the value of the experiment and virtually eliminate typical tunnel dependent bias errors. It should be noted here that it is not sufficient to test in different windtunnels and to use the resulting scatter as an 'overall measurement uncertainty'. One should rather try to understand the origins of the differences. Some more aspects of multi-facility testing will be described in the next section.

2.1.3   CFD validation: a case for multi-facility testing?

There is some uncertainty in windtunnel test results due to various kinds of errors. It is common practise to make a distinction between "instrument error" and "bias error". Instrument errors can be assessed quite accurately. In an AGARD report[5] a methodology is described to quantify these errors in a systematic way. The real problems however are the bias errors that are much more difficult to assess. Bias errors are characterised by a systematic deviation between the measured and the 'true' value, resulting from systematic instrument errors, deficiencies in model shape or 'imperfect' aerodynamic origins. Typical bias errors of an aerodynamic nature are wall and support interference effects, flow quality, transition point uncertainties, flow quality . . .

Even today bias errors limit the possibilities to obtain windtunnel results with the required high absolute accuracy. Testing one model in different windtunnels during a development program sometime raises more questions then can be answered. This is not a very acceptable situation. Flight prediction is most often based on past experience using windtunnel dependent databases. This makes it difficult to test in different windtunnels and limits the information exchange between different partners in a project. To cope with this situation, the aircraft designer takes a pragmatic approach, involving comparative tests in one and the same windtunnel, and the application of corrections based on past experience from windtunnel - flight comparisons. From the point of view of windtunnel test techniques this is even less acceptable. It is essential, if test results are to be guaranteed, to understand the magnitude and the cause of possible bias effects.

Comparative multi-facility tests are well suited to establish bias errors. AGARD has led the way here with standard models (the well known AGARD A, B, C, D and E models[6] that have been tested in various wind tunnels. The results of such comparisons have not always been very satisfactory. Problems with accuracy and the difficulty to assess the origins of certain differences, have been partly responsible for this. However, since that time accuracyies have improved considerably whereas more means are available to assess the origin of particular differences. Such a comparative test would still be to-day very interesting. If such an exercise is to be made, it is essential to limit oneself to a rather simple configuration, easily transferable, with sufficient information to establish in more detail the origins of possible differences (fig. 2.1-1).

How do bias errors in windtunnel test results affect CFD validation? 'Classical' examples are uncertainties in reference conditions, where Mach and/or incidence are adjusted in CFD calculations to improve the agreement with the experiment. For transport type wings with shock waves close to drag divergence, the flow is sensitive for small variations in reference conditions. The drag of transport type configurations is sensitive to wall induced or support induced buoyancy effects, that can easily be an order of magnitude larger then the required accuracy for the designer. Around Mach=1 wave reflections from tunnel walls are known to be problematic. In supersonic flow, flow curvature effects can have a significant effect on pitching moment, even amplified when control surfaces are present. These are a few examples of bias effects that will effect CFD validation.

The simple geometry of a 'standard' model that would be optimal to reveal possible bias effects in a particular windtunnel, would hardly be of interest for CFD validation. If bias errors are established in this way for a particular tunnel it is not obvious how this information can be transferred to a CFD validation experiment. Moreover, if the recommendations given in section 2.1.2 are followed in a CFD validation experiment, some of the experimental uncertainties can be removed or at least be reduced. It is also possible to perform a CFD validation experiment in different windtunnels to establish bias errors. However, in that case an in general much more complicated model will have to be transferred from one tunnel to another. As described in section 2.1.2, complex measuring techniques are often used in CFD validation experiments. Specific experience with a particular technique often resides with the tunnel staff and the applied hardware is difficult to transfer. This argument can be reversed as well. One and the same model can be tested in different facilities such that each of these facilities uses its own specific expertise for specialised measurements. In that case the test can be viewed as complementary with some additional redundancy. A good illustration can be found in[7] where a (large) model was designed (taking into account the tunnel walls) such that a required pressure distribution (corresponding to a 3-dimensional boundary layer close to and beyond separation) was realised. In that particular example, two models of somewhat different scale were manufactured and tested in two different windtunnels (though at the same Reynolds number) using partly redundant and partly complementary test techniques (laser, pressure probes and hot-wires) to measure the boundary layer and wake in detail. But one should realise that such a comparative test requires dedicated teams that have the opportunities to collaborate over a long period of time: it is not sufficient to do tests in two facilities but one should also have the time to analyse and understand possible differences.

2.1.4   Final remarks

The discussion above has clarified that the requirements for 'experimental bias error assessment' and 'CFD validation' are far from identical. The assessment of bias errors requires a specifically designed 'standard' model enables the details of possible differences to be better understood. There is a need for such an exercise from the point of view of an increased emphasis on quality control. Moreover, improvements in measurements techniques and evaluation methods make it possible to deduce clear conclusions from such a comparative test.

However, such a 'standard' model is hardly of interest for CFD validation. For CFD validation one is basically interested in detailed flow field studies to understand possible discrepancies between CFD and experiment. These studies involve expensive and very often tunnel related instrumentation. Bias errors will of course affect the CFD comparison with experiment. They can be reduced by carefully designing the experiment with special attention for reference values and boundary conditions. A comparative test in different facilities is therefore of limited interest, very difficult to do and/or very costly. However, it can have large advantages to test in different windtunnels when this enables the application of complementary (and partly redundant) test techniques for which a particular windtunnel team has its own specific expertise and experience. In that case the quality of the experiment might be increased in a cost-effective way.

6

References:

1. R..G. Bradley: "CFD Validation Philosophy", in Validation of Computational Fluid Dynamics, AGARD CP 437 (1988)
2. W. L. Oberkampf, F.G. Blottner: "Issues in Computational Fluid Dynamics Code Verification and Validation", Sandia Report SAND95-1352 UC-706 (1997)
3. D.P. Aeschliman, W.L. Oberkampf: "Experimental Methodology for Computational Fluid Dynamics Code Validation, ", Sandia Report SAND95-1189 UC-706 (1997)
4. "A Selection of Experimental Test Cases for the Validation of CFD Codes", AGARD-AR-303 (1994)
5. "Quality Assessment for Windtunnel Testing", Report of AGARD FDP WG-15, AGARD-AR-304 (1994)
6. "A Review of measurements on AGARD calibration models", AGARDograph 64 (1961)
7. J.M. Gooden, C. Gleyzes, Y. Maciel: "Experimental study of the flow around 2 scaled 3D swept wings", 28th AIAA Fluid Dynamics Conference, Snowmass, CO, AIAA paper 97-2020 (1997)

Building-block studies for the development of 3-D Navier-Stokes computational codes

	External, perfect-gas, transonic flows			After No. 14, algorithm improvements will be minimal
	Studies	**Steady state data**	**Unsteady data**	**Comments**
1	Hold free stream in all 3 coordinate directions	X		Run free stream for extended number of time steps
2	Low speed (Blasius) boundary layer	X		Compare Cd in all 3 directions
3	Inviscid Busemann biplane (2-D)	X		Supersonic shock crossings and cancellations; compare pressure distributions
4	One-D Shock tube		X	Predict shock and contact discontinuity accurately in time
5	Turbulent flat-plate boundary layer	X		Compare in all 3 coordinate directions: subsonic, transonic, and supersonic
6	Circular-arc bump on a plate (2-D)	X		Geometrically induced separation and reattachment (separation bubble size is critical)
7	Axisymmetric circular-arc bump on longitudinal cylinder	X		Geometrically induced separation and reattachment (separation bubble size is critical)
8	Shock/boundary-layer interaction on flat plate (2-D)	X	X	Usually unsteady; separation and reattachment
9	Transonic 2-D airfoil (attached flow)	X		Multiple angle-of-attack schedule; compare in all three directions
10	Transverse circular cylinder (2-D)	X	X	Broad range of M and Re; compare in all three directions
11	Transonic 2-D airfoil (separated flow)	X	X	Investigate drag sensitivity to geometry, such as leading-edge radius
12	Pitching 2-D airfoil		X	Harmonic oscillation
13	Planar mixing layer	X	X	Convective Mach numbers from ~0 - >1; all directions
14	Transonic/supersonic ogive-cylinder; both high AOA and yaw	X		Cross-flow shocks; separation; vortices (compare symmetry BCs with full 3-D at 0 yaw)
15	Flat plate with 2-D cross-flow jet	X	X	Separation forward of jet and reattachment on jet
16	Multi-element 2-D airfoil	X		Shear-layer/boundary layer interaction
17	3-D Wing (attached flow)	X		
18	3-D Wing (separated flow)	X	X	Investigate separation sensitivity to geometry
19	Wing/LEX at high AOA	X	X	Track vortex shedding and strength
20	Longitudinal cylinder with 3-D cross-flow jet	X	X	Separation forward of jet and reattachment on jet
21	3-D Cavity	X	X	With and without stores in cavity. Deep/shallow cavity. Match sound-pressure levels
22	Fuselage/wing/tail/inlet	X	X	Complex geometry
23	Full low observable aircraft	X	X	

Table 2.1-1 List of 'Building Block' studies

large wing area, high sweep, wing twist for high lift without flow separation at high Mach numbers; fixation permanently attached to wing

simple cylindrical body with ogive nose and straight rear geometry; laid out for combined balance and pressure measurements

horizintal tail with limited pressures, detachable for flow curvature effects

pressures along fuselage and nose to detect pressure gradients and flow angle at nose

pressure section left and right, 3 upper, 3 lower surface pressures on each section to evaluate local flow angle and Mach number differences

straight rear sting, base pressures to be measured

Figure 2.1-1 Sketch of 'standard' model for tunnel comparisons

2.2 The results of a questionnaire

by Dr. G.L. Seibert (USAF/AFRL)

The working group developed a questionnaire that was designed to solicit responses to several questions related to Computational Fluid Dynamics (CFD) code validation and wind tunnel testing of various configurations. The questions were to define areas where the respondents performed their work, with three original choices, namely fighters, transports, and missiles and a fourth, helicopters added later due to some respondents claiming it was a neglected area.

The questionnaire is reproduced in Figure 2.2-1.

RTO/AGARD AVP/FDP Working Group 23

QUESTIONNAIRE FOR CFD VALIDATION

1 IN WHAT AREA'S ARE YOU DEVELOPING CFD CODES FOR AIRCRAFT DESIGN?
_____ transport type _____ fighters _____missiles

2 EXPERIMENTS USED BY YOU FOR CFD VALIDATION (< 5 years)?
☐ "in house" experimental results
☐ data from specific (restricted) collaborative programs
☐ data from the 'open literature'
☐ other (specify):

3 NAME YOUR 3 FAVOURITE EXPERIMENTS FOR CFD VALIDATION

4 HOW ARE YOUR EXPERIENCES WITH CFD VALIDATION?
_____ satisfactory _____ acceptable _____ disappointing
please specify

5 WHAT VALIDATION EXPERIMENT YOU WOULD LIKE TO SEE EXECUTED:
please specify

6 ARE YOU AWARE OF THE 1994 AGARD PUBLICATION AR-303 "A SELECTION OF TEST CASES FOR THE VALIDATION OF CFD CODES"?
_____ no _____ yes

7 IN CASE YOU HAVE OBTAINED THE DATA ON FLOPPY DISK FROM THIS PUBLICATION, WHAT CASES DID YOU USE?
case numbers:_____

8 HOW ARE YOUR EXPERIENCES FOR THESE CASES?
_____ satisfactory _____ acceptable _____ disappointing
please specify:

9 PLEASE GIVE ANY OTHER COMMENTS

NAME (optional):
AFFILIATION:

Figure 2.2-1 Questionnaire for CFD validation

The questionnaire was distributed widely in the US and Europe, including being made available electronically on a web site, but was not met with an overwhelming response. In spite of the small response (32 questionnaires returned) the demographics of the responses were that all three of the original categories were well represented (with, respectively 23, 17 and 15 entries). Many respondents indicated more than one area of interest, which accounts for the aggregate total being greater than the original number of respondents. The listed three possible sources of experimental test cases for CFD validation, (in-house results, specific collaborative programs, open literature) were equally used. The majority (18) of the validators had acceptable experiences with CFD validation, eight found it satisfactory and eight disappointing. A long list of favourite experiments could be made with a few top scores. Noteworthy remarks addressed the need for helicopter code validation and the lack of dialogue between the experimentalist and the CFD user of the data.

The results of the questionnaires were further analysed and then compiled in several ways to help clarify the responses and perhaps indicate where the greatest needs for CFD validation exist.

For the question of what type of experiments were used for CFD validation, it was fairly evenly divided between in-house, specific data and data from the open literature. The "Other" category reflected on the use of theoretical solutions such as flat plates, boundary layers, oblique shocks and shock tube data.

Initially, the request for the three favourite experiments for validation resulted in a master list of 54 different test cases, with some of the same tests being mentioned by several respondents. Initially, these were further broken down into the three categories of fighters, transports and missiles, but it quickly became apparent that many of the test cases were so generic that a case could be made that they would fit all the categories or they could be lumped under the heading of basic research.

Considering CFD validation experiences, most respondents had an acceptable or higher level of comfort with their efforts, although there were many caveats to their comfort level. In general, these had to do with 3D issues, laminar versus turbulent methods, lack of details about the test article geometry, issues with details about the test facility, a lack of enough data from the test, and a better understanding of unsteady flows, among others.

For the question about what validation experiment they would like to see executed, it quickly became apparent that more complex test cases were needed as the capability of the CFD codes has developed over the years. Specifically, there was a stated need for high lift cases, including multi-element airfoils; unsteady tests of various configurations; detailed measurements of attached and separated flows over a wide range of Reynolds numbers; tests that include aeroelastic effects; off-body measurements; detailed experiments on turbulent and transitional flow and there was interest in rotary wing flows for helicopter applications.

A significant number of respondents were not aware of the two volume set of the 1994 AGARD Publication AR303, "A Selection of Test Cases for the Validation of CFD Codes." [1]. Volume I discusses the CFD requirements for code validation, the requirements for experiments for CFD validation, introduces the data and then gives summaries of the test cases. Volume II presents, in detail, 39 experiments in various categories and configurations with, theoretically, enough information to validate a code. Included in Volume II is a procedure for obtaining and using floppy disks with the complete set of data from all the tests. Of those respondents who were familiar with the report, only a few reported that they had obtained the floppy disks and their experiences were mixed. Unfortunately, they did not explain in detail the basis for their comments so it is not clear as to what was acceptable data and what was not.

The general comments re-emphasised the need for more tests involving unsteady flows, 3D flows and, generally, more complex flows. It was also noted that for true validation testing, a CFD specialist should be involved with the project team from the beginning.

A further review of the questionnaire responses was done to classify them more carefully with respect to the usage of AGARD AR 303 and to characterise the reported experiments more closely within the three categories, namely fighters, transports and missiles. Additionally, since many of the responses were more generic, a fourth category, basic research was added.

An important feature of the referenced experiments is the date they were conducted, since it is valuable to know the state-of-the-art of computational fluid dynamics when the experiment occurred. As the science of CFD has matured, the demands on data have become more exacting, which limits to some extent the usefulness of older data.

In the case of fighters, there were only two specific references that were clearly in that category and these did not lend themselves to further analysis. This was also the case for missiles and for the most part, the tests that were more identifiable as basic research. As a result, it was determined not to create a chart for the dating of the referenced material in those three areas, therefore only the one chart was created which includes some aspects of the other areas.

Table 2.2-1 refers to the actual dates of the aircraft type tests done that were reported by the respondents and extracted from the questionnaires. The dates as listed are as close to the actual dates as could be determined from among the questionnaires, the AGARD reports and from discussions with members of the working group. From the table it can be seen that the tests reported cover the period from the 70's to the middle 90's. Only a few are actually in the AGARD reports previously mentioned in the original questionnaire. Clearly, the age of the test was not the determining factor in its use by the modern CFD researcher. What was not asked in the questionnaire was what was the defining reason for using that particular test case for the validation computation. This is an area for further review and discussion.

References:
1. *A selection of Experimental Test Cases for the Validation of CFD Codes. AGARD-AR-303 Vol I and II*

Table 2.2-1 Referenced Generic Test Cases in Returned Questionnaires

No	Experiment	Date	Source
1	2D Airfoils in Transonic Flow (NACA 0012)	1970's	
2	RAE 2822	1970's	AR 138
3	Wing on a wall ONERA M-6	1970's	AR 303
4	2D Airfoil with high lift devices: NLR 7301 with flap	1975	AR 303
5	2D Airfoil with high lift devices: Garteur A-310 High Lift Experiment	~1980	
6	2D RAE 5225 Airfoil	1980's	
7	International Vortex Flow Experiment	1980's	AR 303
8	DLR F4 Wing/Body	1980's	AR 303
9	Wing on a Wall: Wing C	1980's	AR 255
10	AS-28G Wing/Body/Pylon/Nacelle configuration	~1985	
11	DLR F-5 Wing	1985	
12	Garteur European Shear Flow Experiment	1990	
13	ONERA A-340 Wing-Body Tail Exp.	1990's	
14	Multi-Element A-320 Profile		
15	NLR 7702 TE-study	1990's	AR 303
16	ELAC Delta Wing	1995	

3. NEEDS AND POSSIBILITIES FOR CFD VALIDATION THROUGH MULTIFACILITY COLLABORATIVE WINDTUNNEL TESTING

3.1 Fighters

by Dr. J.L. Fulker (DERA) and Dr. J.H. Fox (Sverdrup Inc.)

3.1.1 Present difficulties in obtaining test cases

Typically, CFD developers and designers focus on different validation issues. Code developers are comfortable with a mix of component (building block) test cases from various sources such as AR-303[1], unpublished proprietary experiments, and (usually restricted) experiments run at the various national laboratories. However, designers desire CFD validation using test cases based on modern fighter configurations.

All freely available test cases are of configurations representative of early generations of design[1,2,3]. Those designs sought optimum aerodynamic performance. But designs of current and future configurations for fighter aircraft are driven by other considerations, such as minimal radar cross-section and low IR signature. Thus, new configurations are radically different from past designs. They exhibit aligned aero surfaces, low profiles, faceted surface contours, reaction-jet control, and submerged engine inlets and exhaust nozzles. Since designers require CFD validation data from specific configurations that are closely related to those on which they are working, the currently available test cases are woefully inadequate.

If a multi-national test case could be agreed to, it would allow CFD validation for the design community to be put on a firmer footing. A major stumbling block here is the availability of a suitable model. It is conceivable that one group could be funded to design a generic model. However, if an airframe company were contracted, they probably would not include their latest technology. Another possibility is that an RTA/AVT study group could carry out the design and manufacture of a suitable windtunnel model. But that could be accomplished only with great difficulty, because of RTA's having little experience in this area. But even if a windtunnel model were available for a modern configuration, it is doubtful that one configuration would exercise all the desired physics needed for validation. So, the best outcome would be a suitable set of generic models becoming available from a government-sponsored programme. Until this occurs, the design community is dependent on geometry and data released by manufacturers (with the concurrence of the nation's defence establishment) on their latest models when they deem it safe to do so. But data released by manufacturers, though welcomed, and despite a lot of experimental effort, tend to be very limited. The data tend to be restricted to overall forces and pressures on lifting surfaces, with little or no flow visualisation or flow-field measurements. This is usually the result of a need to reduce the cost of testing. But it could also be the result of the fact that the manufacturers' flow physics experiments are done privately and go unpublished.

3.1.2 Need for comprehensive multi-facility experiments

In spite of the aforementioned difficulties, there still exists a need for new experiments, however they are obtained. And these experiments must include more than the usual surface pressures and force and moment characteristics, for these are clearly insufficient for validating new CFD codes. Further details of the flow should be available in order to show that the code is modelling the flow correctly.

Along with the conventional data, additional data that should be available is as follows:

- More detailed surface pressure measurements, especially in difficult areas such as corner flows and tip flows
- Detailed off-surface flow measurements of at least all three velocity components along with some form of visualisation of flow-field features, so that one could readily determine:

- path, strength, and lateral extent of major vortices
- details of induced separation and reattachment zones
- wake properties from all configuration components.

The desirability of making this array of measurements leads naturally to multi-facility experiments using more than one model. However, the use of expensive and complex test techniques may be better suited to certain facilities. By splitting the load and cost across at least two facilities with the required test techniques, it is much more likely that a full data set encompassing the desired measurements would be obtained.

3.1.3   Potential test cases

Although there is a clearly defined case for conducting a multi-facility CFD validation experiment, it is conspicuous that there are political and proprietary barriers to obtaining a suitable model for testing within a multi-national environment. Therefore it can only be concluded that for the foreseeable future testing of a set of configurations of this kind is out of the question. However, the possibility arises of conducting, complex, building block experiments to obtain, at least, some of the required information and this needs to be explored further within the International community.

References:

1. *A selection of Experimental Test Cases for the Validation of CFD Codes. AGARD-AR-303 Vol I and II*
2. *Fisher, D. F., "F-18 High Alpha Research Vehicle Surface Pressures: Initial In-Flight Results and Correlation With Flow Visualization and Wind-Tunnel Data," AIAA Paper 90-3018, Portland, Oregon, August, 1990.*
3. *Erickson, G. E., Hull, R. M., Banks, D. W., Del Frate, J. H., Schreiner, J. A., Hanley, R. J., and Pulley, C. T., "Experimental Investigation of the F/A-18 Vortex Flow at Subsonic Through Transonic Speeds, Invited Paper," AIAA 89-2222, July, 1989.*
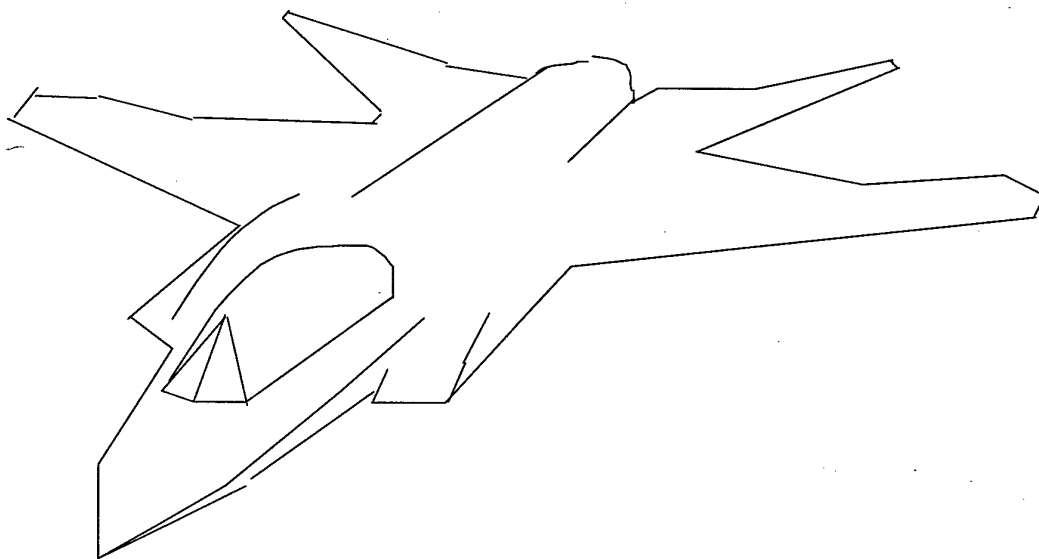
Figure 3.1-1 Generic Modern Fighter Configuration

3.2   Transport type aircraft

by Dr. G. Redeker (DLR) and Dr. A. Amendola (CIRA)

3.2.1   Introduction

For civil transport two types of aircraft configurations are of interest:
- The transonic transport aircraft in various sizes with cruise Mach numbers $0.75 < M < 0.85$ equipped with turbofan engines, and
- The propeller driven aircraft at moderate flight velocities as a small Regio-Liner.

For military application future transport aircraft configurations are under discussions which are also propeller driven.

Three types of validation experiments can be derived from the aircraft configurations under consideration as candidates:
- Transonic transport aircraft in cruise flight condition,
- Transonic transport aircraft in high lift condition, and
- Propeller driven aircraft at moderate flight velocities.

For the cruise flight condition there seems to be no need for a further validation experiment. This is to a large extent covered by running programs in Europe under the auspices of GARTEUR (a co-operation between among others the aeronautical research institutes in Europe) and more specifically in the action group AD(AG-28) "Transonic wing/body code validation experiment". For the aspects of engine/wing integration the BRITE EURAM programs ENIFAIR and AIRDATA as sponsored by the European Union (EU) are to be mentioned. The needs for a further validation experiment for propeller driven aircraft will be partly covered by experiments in execution in the APIAN program of BRITE/EURAM. As comparable programmes for the transonic transport in high-lift conditions are missing at present[1], this configuration was selected as a potential candidate.

3.2.2   Review of available cases for a high-lift configuration

For this type of configuration the following test cases have been found in literature:
- ALVAST-Model measurements; results gained in DLR programmes, DLR/NLR cooperative programmes and in the EU-programmes DUPRIN I/II and ENIFAIR. Not all results are available.
- B 737-100-Model measurements; results gained by NASA in the TSRV-programme. Results are not open, except published examples.

The ALVAST-model owned by DLR[1] is a transport type aircraft in high-lift configuration, Fig. 3.2-1. The model is built up by a high-lift wing with slats and flaps, pylons, model engines and a fuselage. If can be used as complete or as a half model. Engine simulators of different kinds have been used and exist: Throughflow nacelles (TFN), TPS engine simulators of various bypass ratios (TF, VHBR, UHBR). The model is equipped with nine pressure stations on the wing including taps on slats and flaps and several rows of pressure taps on the fuselage. Measurements of surface pressures on wing and fuselage as well as aerodynamic forces and pitching moments have been performed. No further flow details have been measured.

The B 737-100-model, owned by NASA[2] is the wind tunnel model of a complete B 737-100 research aircraft flown in the NASA TSRV-programme. The wind tunnel model is shown in Fig. 3.2-2. The configuration consists of high-lift wing, pylon, throughflow nacelle, fuselage and empennage. Wing surface pressure at 7 spanwise stations including slats and flaps have been measured and forces and moments. No further details have been investigated.

Both cases exhibit further detailed information on flow field data needed for CFD-validation purposes.

---

[1] in the 5th Frame Work Programme high lift proposals might be included

3.2.3   Needs for new multifacility experiments

From the viewpoint of CFD-validation for high-lift configurations new validation experiments are highly desirable. Surface pressures, force and moment coefficients are not sufficient for validating new flow solver codes. Further details of the flow should be available in order to prove if the code is modelling the flow correctly.
Besides the already existing data of
-    force and moment coefficients
-    surface pressures,
the following data should be available
-    velocity profiles in the boundary layer of different wing elements
-    shear stress of the flow for turbulence modelling
-    details of separation zones on wing elements (on surfaces or/and on sharp edges like slat lower surface and flaps housing)
-    wake of wing including vortex paths of wing tips and flap side edges

Multifacility experiments are necessary due to the requirement of expensive and complex test techniques as PIV and PSP or automatically controlled probe measurements. Splitting the load on several or at least two wind tunnels it seems to be easier to get the complete set of data.
A further problem could be solved by multifacility wind tunnel testing. The problem of wind tunnel wall interference and the sting correction problem can be tackled easier if there are data sets of different wind tunnels available.

3.2.4   Candidate test cases

It is proposed to use either the ALVAST-model or the B 737-100-model for the new validation experiments. Both models exist in a high-lift configuration and are about of equal size.
The ALVAST-model has the advantage that it could be made available for new tests, whereas the situation for the B 737-100-model is unclear at the moment.
On contrary the B 737-100-model has the advantage that flight test data are available which – if released – could also be used for code validation.
The following flow physics should be studied:
-    boundary layer development on various wing elements; confluent boundary layers
-    influence of transition fixing on high-lift development
-    separation zones on wing elements; especially trailing edge separation on the flaps and separation zones in the slat cove and in the flap housing
-    flow fields in the gap between the wing elements
-    wake development and vortex formation from wing tips and flap side edges
Advanced measurement techniques have to be applied in order to get the necessary information. It is envisaged to use:
-    PSP-technique for a complete survey of surface pressure
-    PIV-technique to get velocity profiles in the boundary layers and in the separation zones
-    PIV-technique for wake measurements and determination of vortex locations

Candidate facilities for the tests are the DNW on the European side and the NASA LRC 14 ft x 22 ft wind tunnel on the US side. Both models have already been tested in their respective wind tunnel.

References:

1.   R. Kiock: The ALVAST Model: Engine Simulation on a Transport Aircraft Model Proceedings CEAS European Forum Wind Tunnels and Wind Tunnel Test Techniques Cambridge 1997, pp. 13.1 to 13.14
2.   D.C. Edge, J. N. Perkins: Three-dimensional Aerodynamic Analysis of a Subsonic Transport High-Lift Configuration and Comparisons with Wind Tunnel Test Results AIAA-95 1846-CP, 1995
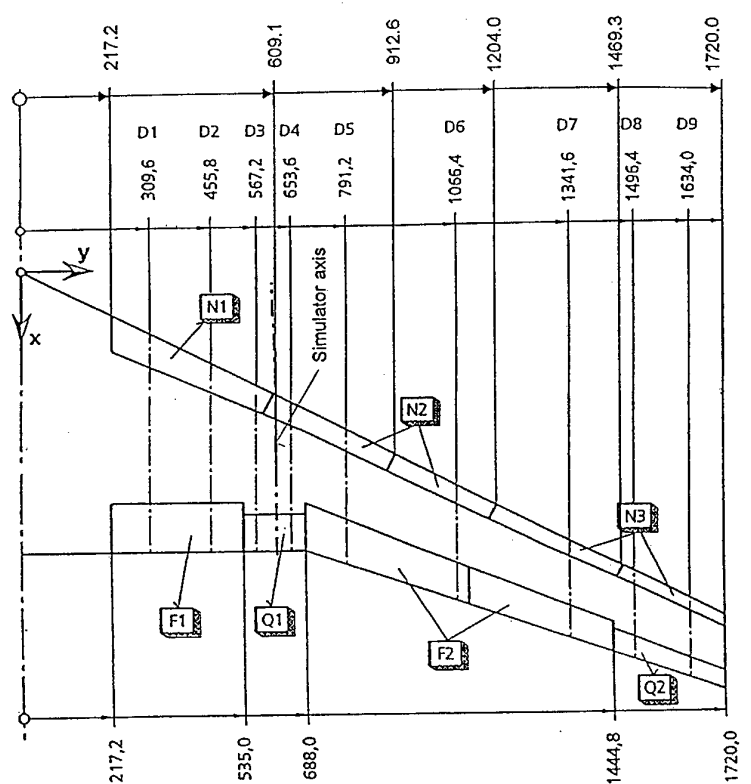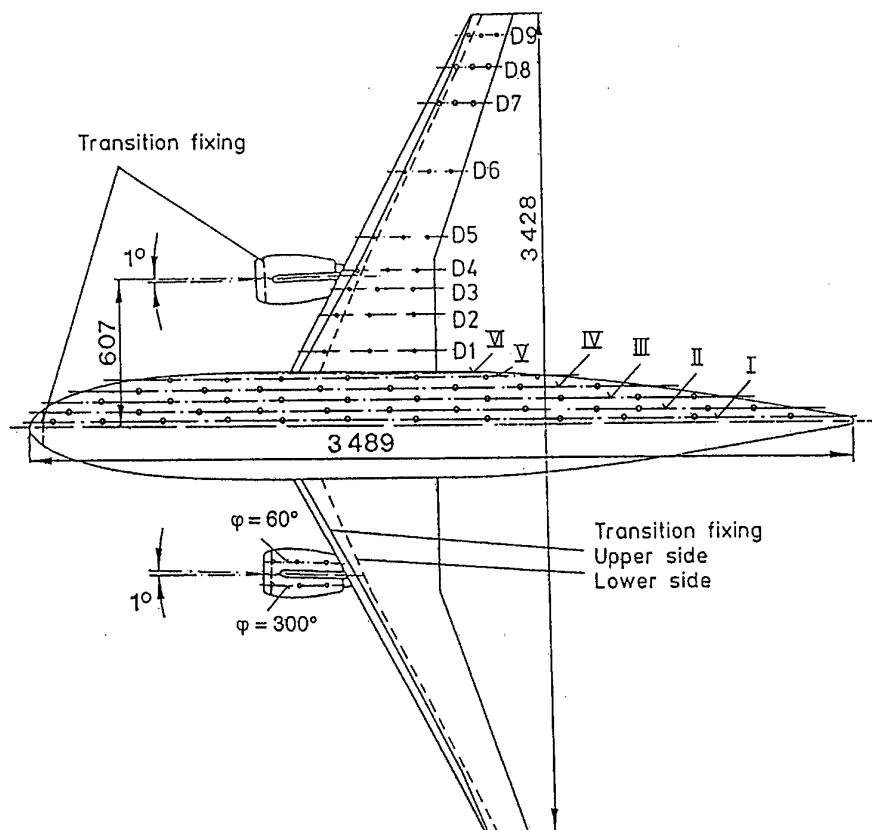
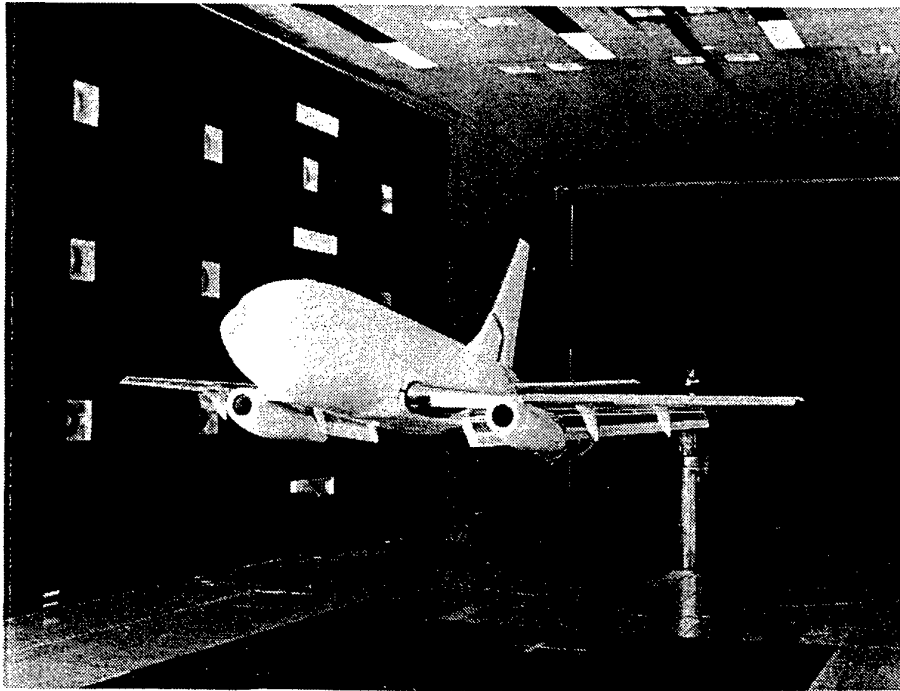**Figure 3.2-1 ALVAST wing/body configuration in high-lift condition**

Engine
℄

Flush Pressure Taps

η = 0.23

η = 0.40

η = 0.54

η = 0.58

η = 0.71

η = 0.84

η = 0.90
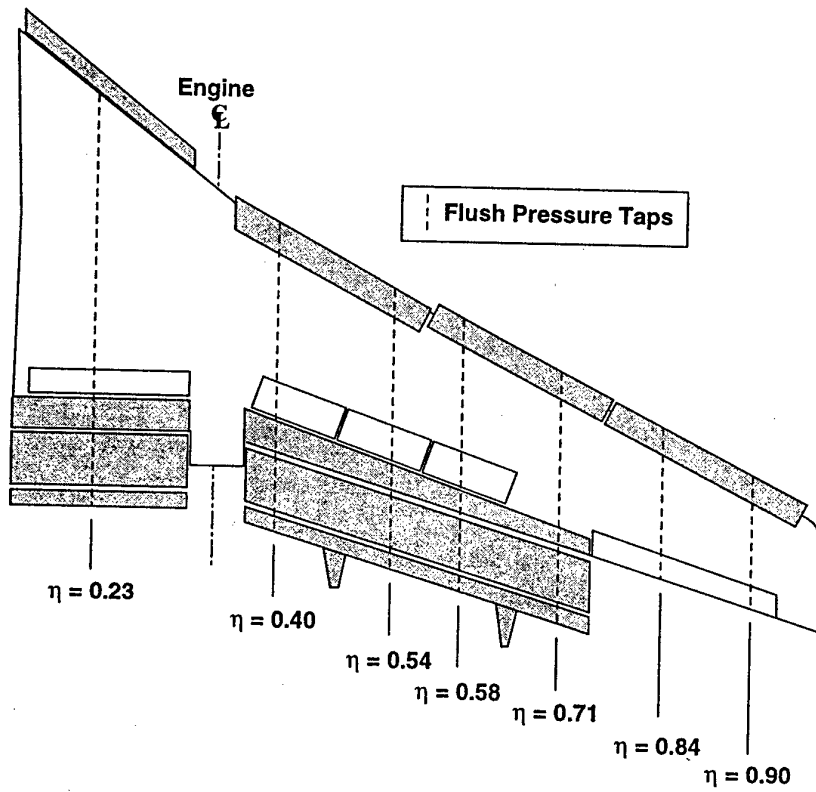
Figure 3.2-2 Wind tunnel model of B737-100 in high-lift condition[2]

3.3   Missiles

by Dr. M. Khalid (NRC/IAR) and V. Schmitt (ONERA)

3.3.1   Introduction

As part of a feasibility exercise, a number of experimental test cases have been retrieved from the AGARD Report 303 [1] and other literature sources to assess the suitability of the experimental results for CFD code validation. The intention is to determine from these studies the type and detail of a future wind tunnel experiment, which will satisfactorily address the requirements of a comprehensive code validation exercise. Arriving at an appropriate missile configuration, which will challenge the scope of modern CFD codes in areas of multi-blocking, multi and adaptive gridding for viscous resolutions as well as testing the accuracy of turbulence models in separated flows would be another dimension of the same task.

3.3.2 A Survey of Existing Test Cases

It would be rather tedious to give a detailed blow by blow account of all the test cases existing in literature. Instead it would be more suitable to present a brief description of the test including a description of the configuration together with the physics of the flow and the range of test conditions investigated. The comments would also include the suitability of the existing configuration and the associated wind tunnel measurements as a candidate for code validation. Where appropriate the case identification for various configurations is retained as it appears in the original reference. The first Group A deals with blunt nosed cylindrical bodies. The ogive-cylinder type simple configurations are treated separately in Group B. Configurations with non-circular missile bodies are contained in Group C. Group D deals with missiles with conventional wings/control surfaces and grid fins.

Blunt Nosed Simple Missiles- Group A
The missile configurations in this group (see Table 3.3-1) have typically blunt nosed leading edges followed by stubby cylindrical after bodies. These configurations at low speeds are particularly well suited for boundary layer studies. At high Mach numbers the flow past blunt nosed shapes can give rise to complex shock patterns and may have application in heat transfer studies. The cases identified here contain detailed measurements of boundary layer growth, transition and separation at low speed. The experimental data also include identification of vortices in terms of pressure measurements at various axial stations. Other useful data for code validation include the surface pressure distribution measurements, velocity and shear stress profiles as well as flow visualisation records. This type of measurements would be appropriate for CFD code validation in viscous regimes and examining the accuracy of the turbulence models.

Ogive- Cylinder Missiles- Group B
There is a large variety of experimental data available for this class of missiles. Most cases considered in this group (see Table 3.3-1) deal with supersonic flow, where the interest amongst other parameters is also focused on various shock patterns captured using schlieren photography. For the configurations investigated, at M=1.5 and M=2, the angle of incidence is ranged from 0 to 23°. For lower Mach number cases of the configuration the angle of attack range is extended up to 90°. This is an important area of research because higher angles of incidence provoke earlier separation, which is essentially a 3D phenomenon leading to interesting vortex development along the length of the missile. In most cases the measurements also include surface and flow field pressure measurements, boundary layer growth measurements including separation leading to the evolution of vortices. The data would be well suited for subsonic-supersonic surface and flow field data validation as well as the prediction of viscous characteristics and vortex formation.
In addition to the measurements reported[1] for the ogive – cylinder type configuration there are other experiments reported in literature for similar configuration. The Technical Cooperation Program (TTCP) only recently carried out a code validation study of an ogive-cylinder missile tested in the DERA, Bedford, UK tunnel[2]. The model tested is a 3-calibre ogive with a 10-calibre cylindrical after

body. The data included surface and flow field pressure distribution measurements, pitot surveys, which showed the various separation regimes including vortices development, and force balance measurements. Shock effects were also recorded through schlieren pictures. One of the present authors[3] too has used this configuration for some in house code validation.

Attraghji in references[4,5] presents wind tunnel results on a family of cone-cylinder bodies of revolution at subsonic (M=0.5) and supersonic (M=1.5 and M=2) Mach numbers. The data included surface pressure distribution as well as balance force measurements up to 25° angle of incidence. Flow visualisation records were made at M= 3.5 and a novel method of surface shear stress evaluation was proposed, which used an oil dot technique. The family of nose profiles investigated is described by $r/r_{rmax} = 1-(1-x/LN)^m$, where x is the axial length and LN refers to the length of the nose cone, and m= 1, 4/3, 5/3 and 2 were used with a nose fineness ratio of $LN/2r_{max}$. The values of m were such selected to give roughly equal interval of apex angle.

Non-Circular Missiles - Group C

The experimental data for this category of missile bodies would be appropriate for viscous code validation especially for investigation of boundary layer separation and vortex prediction. The configuration considered (see Table 3.3-1) appears to show relatively large regions of separated flow even at small angles of attack, which eliminates the risk of flow contamination through blockage effects[1]. This would provide a meaningful challenge for the CFD codes to predict separated flow regions on a suitably defined simple configuration. Other useful data for code validation include surface pressure measurement and integrated forces.

Missiles with Wings/Canards/Control Surfaces and Grid Fins- Group D

There is large variety of missiles, which can qualify to be included in this group. In most cases the additional surfaces are attached to the missile body for providing better stability and control through the flight trajectory envelopes. Usually such demands are placed when the missile is accurately guided towards a given target or has to attain a certain range within a prescribed trajectory. For example there are missiles supporting only control fins, or control fins with wings for generating higher lift albeit with some additional drag penalties. Some missiles are equipped with canards for enhanced performance in pitch.

The grid fins are some of the novel and latest control surface devices which if appropriately designed could produce better lift and roll performance in comparison to the planar surface control devices. An example is given in [6]. Their shorter spanwise dimension relative to the planar fin and 'honey comb' structure enable them to withstand relatively higher bending and twisting moments. They do however impose a severe penalty in terms of the drag, which too may not always be such an unattractive feature when the intent is to decelerate a store towards a target when it is released from a fast moving platform.

Within same requirements of accurate trajectory performance, sometimes a missile is equipped with a capability of lateral thrust[7,8]. This additional vectored thrust is obtained through a high-speed jet nozzle installed at a strategic location along the length of the missile. The intensity of the jet flow and its vector angle can provide fine trajectory adjustments for the path alignment.

Another type of missiles, which merits attention, is the missile with intakes. Different types and number of intake installations are given in [9] which provides a discussion on the importance of intakes for certain design of missiles. The number and design of the intakes takes into account the following factors:

- internal performance as thrust and specific impulse
- external aerodynamic characteristics as drag and lift-to-drag ratio
- operational constraints such as overall dimensions, which must also account of store carriage and radar signatures.

Some basic aerodynamic performance is also affected by the intakes. The lift is usually increased mainly because of the length of the intake nacelle which will also change the center of pressure. The drag too is increased by as much as 38 % of the total drag for a four axisymmetric intake installation at M=2.0.

3.3.3 Recommendations for Future Tests

A future wind tunnel test must be defined with an aim to investigate all the important flow complexities known to exist in flow fields past missile type configurations. Once these flow complexities are correctly quantified and measured, they would then present the appropriate challenge to the CFD practitioners to simulate them in a comprehensive numerical exercise. The flow field characteristics, which are of primary interest to the missile designers, include:

- Boundary layer effects including separation and the evolution of vortices.
- Angle of attack effects including transition studies.
- Plume effects at different angles of attack
- Interference effects between missile bodies/wings/control surfaces.
- Lateral jet effects on the missile flow field.
- Air intake design and flow field studies.
- Non-circular geometries for missile bodies.

Any configuration selected for wind tunnel testing and subsequent CFD code validation exercise, which attempts to encompass all these attributes, might become unacceptably sophisticated in form and prohibitively expensive for numerical simulation. It might then be an appropriate recourse to select a missile configuration, which may not address all the issues in question, but would provide the basis for testing all the critical methodologies of a CFD code.

Towards that end, a missile such as Aerospatiale's 'ASTER'[8] would be quite appropriate for such a study. Because this configuration is for obvious reasons not available one may resort to generic models. Three candidate configurations for multi-facility testing are shown in Figures 3.3-1 to 3.3-3. These models exist or could be easily manufactured and would represent fairly standard cases for future references. Additional information on the two latter configurations may be obtained from [9,10].

As mentioned earlier the wind tunnel tests should include:
- force and moment measurements by balances ;
- surface pressure measurements in both axial (wings) and azimuthal directions (fuselage) ;
- flow visualisation records to study shock effects ;
- velocity profile and/or flowfield measurements at strategic axial locations, especially with an aim to trace the development of vortices. This data would also be used for transition studies and separation at higher angles of attack.

The wind tunnel tests could be conducted in any of the following tunnels:
- ONERA FR Wind Tunnels ;
- DERA UK Tunnels ;
- NASA, AEDC, Boeing US Tunnels;
- IAR/NCR CA High Reynolds Number Wind Tunnel.
- DNW (GE/NL) windtunnels

The proposed configurations will provide a challenge in multi-block mesh topologies and multi gridding for efficient running. The verification of boundary layer properties correctly would require the implementations of finer grids or appropriate grid adaptation to resolve the sub scale viscous flows. Being able to predict the separated and/or vortical flows at higher angles of attack would necessitate the implementation of higher order turbulence models.

3.3.4 Conclusion

The study shows a wide variety of missile configurations, which have been tested in various facilities throughout the world. Only a handful of the cases were selected for the present investigation. The measurements have ranged in Mach numbers from subsonic, transonic to supersonic flow conditions. In most cases the investigations have involved surface pressure and/or balance force

measurements. The flow field data in viscous and / or inviscid regimes is not as abundant. The depth and quality of data collected seems to decrease as the sophistication of the missile configuration increases.

Occasionally the tests in tunnels have been reported without any clear statement about the accuracy of the measurements including correction aspects. If it is not possible to complete the existing information this would make the prospect of meaningful CFD code validation somewhat questionable. Only in a few cases implying multi-facility testing have been mentioned. In order to delete this lack at least 3 missile configurations have been identified as possible candidates for future wind tunnel experiments. However, the justification for multi-facility testing has to be substantiated.

References

1. *A Selection of Experimental Test Cases for the Validation of CFD Codes. AGARD-AR-303 Vol. I. and II.*
2. *Sturek, Walter, B. ' The Application of CFD to the Prediction of Missile Body Vortices', AIAA 97-0937, 35th Aerospace Sciences Meeting & Exhibits January 6-10, 1997/ Reno, NV.*
3. *Khalid, M. and Sun, Y., 'Computation of flows past grid fin Missiles', RTO/AGARD Symposium on "Missile Aerodynamics", 11-14 May 1998, Sorrento, Italy.*
4. *Attraghi, E. G. 'Pressure Distribution Over a Family of Inclined Long Slender Bodies of Revolution at M=0.5, 2.0 and 3.5', August 1968 LTR-HA-5X5/0029, National Aeronautical Establishment/NRC Canada.*
5. *Attraghi, E. G. 'The Influence of Mach Number, Reynolds Number, Semi-Nose Angle and Roll Rate on the Development of the Forces and Moments Over a Series of Long Slender Bodies of Revolution at M=0.5, 2.0 and 3.5', Sep. 1968 LTR-HA-5X5/0020, National Aeronautical Establishment/NRC Canada*
6. *Washington, W., D., 'Grid Fins- A New Concept for Missile Stability and Control', 31st, Aerospace Sciences Meeting &Exhibits AIAA 93-0035, Jan. 11-14, 1993/Reno, NV.*
7. *Srivastava, B. 'Lateral Jet Control of a Supersonic Missile: CFD Predictions and Comparisons to Force and Moment Measurements', 35th, Aerospace Sciences Meeting and Exhibits January 6-10, 1997/Reno, NV.*
8. *Dormieux, M. and Mahe, C. 'Calculs Tridimensionnels De L'Interaction D'un Jet Lateral avec un Ecoulement Supersonique Externe', AGARD CP 437 Vol. 1-2., 1988.*
9. *Special Course on "Missile Aerodynamics", AGARD-R-804, June 1994.*
10. *Graves, E. B., 'Aerodynamic Characteristics of a mono planar Missile Concept with Bodies of Circular and elliptical Cross Sections', NASA TM-74079, 1977.*

Table 3.3-1 Selection of Test Cases from Reference 1

Configuration	Flow Conditions	Facility
Group A		
Prolate spheroid Test case C2 [1]	M = 0.16, 0.13, 0.23 α=10° and 30° R/d =7.7X10^6, 6.5X10^6, 43X10^6	3m X 3m Low-Speed (NWG) wind tunnel at DLR Göttingen 4.5m X 3.5m F1 wind tunnel at ONERA Fauga-Mauzac
Ellipsoid-Cylinder Model Test case C4 [1]	V= 50 m/s α=-20° to 30° R =5.6X10^6 Based on body length	1.8m X 1.4 m F2 low speed wind tunnel at the ONERA Fauga-Mauzac Centre. The tunnel has solid walls.
Group B		
Power Law Nose Circular Body Test case C1 [1]	M = 1.5, 23° $\leq \alpha \leq$ 8° R/d = 1.2 X 10^6	0.6mX0.6m (Solid Walls) TMK wind tunnel at DLR Köln-Porz
Ogive-Cylinder missile body Test case C5 [1]	M = 2. α= 5° to 20° R/D =0.16X10^6	0.3m X 0.3m ONERA S5CH (Solid Walls) Tunnel
Ogive Nosed Slender Body Test case C3 [1]	V=0.1m/s to 60m/s α=-5° and 90° R/d =0.005X10^6 to 0.765X10^6	3m X 3 m Low-Speed open jet (NWG) wind tunnel at DLR Göttingen 3.25m X 2.8 m Low-Speed (NWB) open jet wind tunnel at DLR Braunschweig
Group C		
Non-Circular missile body Test case C6 [1]	M = 0.4, 0.9, 2.,4.5 α= 0°, 10°, 20° R/D =12.5X10^6 to 20.5X10^6	0.76 m X 1.75 m ONERA S2MA perforated wall tunnel, with2.9% open area ratio; sub, super and transonic conditions 0.8m X 0.76 ONERA S3MA solid wall tunnel- supersonic flow
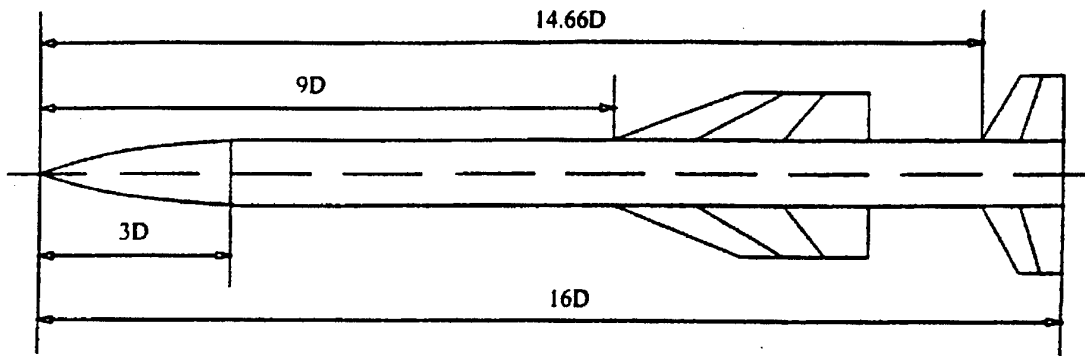
Figure 3.3-1 Generic missile configuration used at ONERA
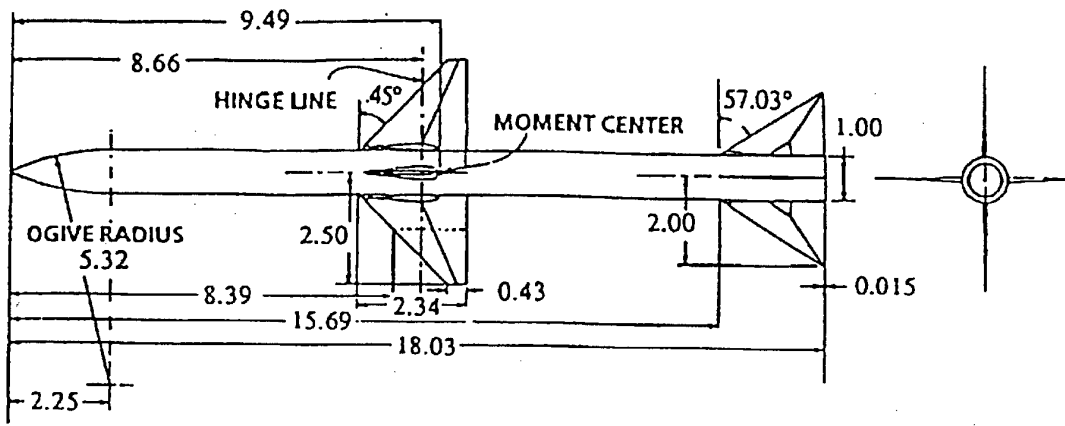
Figure 3.3-2 Air-to-air missile configuration used in validation process
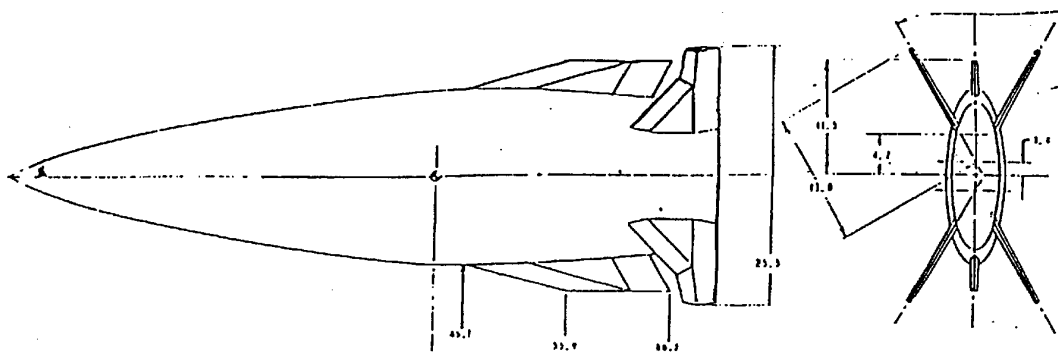
Figure 3.3-3 Non-circular cross section missile with wings and fins

3.4 CFD code validation for rotorcraft flow field analysis

by Dr. A. Amendola (CIRA) and Dr. M. Khalid (NRC/IAR)

3.4.1 Helicopter Aerodynamics: General Aspects

The flow field past a helicopter is very complex, consisting of such phenomena as unsteady rotor wake effects, vortices and large regions of separated flows. Additionally, there are flow interactions between main rotor and helicopter fuselage and boom, as well as flow impingement effects from helicopter body to the tail rotor. All these flow patterns are quite intractable individually and produce a daunting flow field picture when considered ensemble. These flow complexities are present with varying degrees of sophistication during different helicopter manoeuvres which may at a given time involve one or combination of such movements as hover, climb, descent, cruise and lateral sideways motions.

Perhaps the most dominant flow features in a helicopter flow field originate from:
- rotor wake
- interaction effects between components
- flow unsteadiness
- body bluntness
- blade/vortex interaction.

When analysing the helicopter flow field, we primarily concern ourselves with the aforementioned aerodynamic characteristics and ignore other aeroelastic, structural or inertial implications of loading or surface deformations, the inclusion of which, would certainly increase the complexity manifold. In passing then, we may observe that the aerodynamic study of rotorcraft flow field is one of the most challenging issues facing aerodynamicists from both experimental and computational points of view.

Interactional Aerodynamics
The flow interactions between different components of helicopter is of paramount importance and need to be appropriately addressed when evaluating overall aerodynamic loads. Interactional aerodynamics is generally classified according to the configuration elements that are involved, with the most important being:
- main rotor/fuselage
- main rotor/tail boom
- main rotor/tail rotor
- main rotor/tail plane

The static and dynamic effects associated with these different types of interference have important consequence on flight mechanics characteristics and handling qualities and have to be suitably accounted for when designing feed back and control mechanisms.

At low speed, examples of important phenomena owing to flow interference include:
- vehicle pitch up during transition from hover to cruise flight
- control problems and extreme bank angles during sideward flight in and out of ground effects
- control problems during quartering flight.

Other important effects, which are prevalent over larger domains of the helicopter flight envelope include:
- tail shake provoked by interference between hub/pylon wake and helicopter tail planes
- tail rotor noise that is heavily influenced by the main rotor wake
- dynamic stability

Flow Unsteadiness
The unsteadiness of the flow is intrinsically connected with the mechanism by which helicopter rotor creates lift, that is a wing that rotates at relatively high speeds and carries a fixed blunt body. The relative motion between rotating and non-rotating components of the helicopter induces primarily unsteadiness in the flow. The complex wake emanating from the rotor also induces unsteadiness, as an

upcoming rotor blade interacts with the wake of preceding blade, or when the helical rotor wake down-washes upon the main fuselage.

Other aerodynamic interactions, which also warrant attention originate from the tail rotor which rotates at high speeds to provide necessary stability to the helicopter. The tail rotor may be operating partially downstream of the rotor hub and partially in regions aft of main rotor and fuselage wakes, thereby creating a situation in which three wakes interact strongly in an extremely complex vortical and unsteady flow field. The resulting flow field has important consequences for tail rotor performance and aft empennage loading.

Body Bluntness

Helicopter fuselage is generally an extremely bluffed body shape, which is prone to flow separation. The fuselage may support regions of large-scale flow separations in the fore regions of the nose even before the canopy. This flow behaviour, together with traditional flow separations that are found in the rear part of the fuselage or in the dog-house regions below the rotor can challenge best of the turbulence model that CFD has to offer. It is not unusual to come across laminar flow separation in the fore regions of the fuselage leading to the formation of a separation bubble in the nose regions of the fuselage.

Flow separation studies are therefore an important aspect of the field around typical shapes of many helicopters currently in operation and being designed for the future. Until recently fuselage aeodynamics had low priority in helicopter design, which is in sharp contrast to fixed wing aerodynamic practice, where flow separation suppression is one of the main design preoccupation for efficient flight. In addition, flow separation studies are equally important for military helicopters which are often flown with mission kits (emergency equipment, fuel pods, cameras, rescue equipment) attached to the basic configuration and may be responsible for even more flow complexities and separations.

Blade/vortex interaction (BVI)

For a helicopter rotor (main rotor) operating in certain low speed partial-power descent flight conditions, the upflow tends to force the rotor wake (blade tip vortices) into the rotor disk plane, causing strong blade-vortex interaction (Blade-Vortex-Interaction). In this case the unsteady loads caused by the strong induced effects of the tip vortex during its brief but repetitive (periodic) close encounter with the blade, generate impulsive noise. Therefore there is an interest in the aerodynamic and aeroelasticity-structure community to study the vortex emanating from a given blade and mechanisms installed on an approaching rotor blade to suppress or breakup the vortex. Apparently, this can lead to alleviating blade vortex interaction and may provide some noise suppression.

3.4.2   State-of-the-art of experiment

In the following sections short descriptions of some relevant experiments suitable for validation purposes are given. The experiments described here address the following helicopter components:
-    isolated rotor
-    isolated fuselage
-    rotor and fuselage
-    complete helicopter configuration

Before considering the various experiments, perhaps it would be worthwhile to reiterate that the complexity of the flow fields makes the prospects of experimental or numerical simulation equally challenging. In the wind-tunnel environment for proper simulation, the aerodynamic effects need to be considered along side with the elastic and inertial behaviour of helicopter components in order to achieve similitude in both aerodynamics and the structural dynamics of the blades. The complexity of the flows past helicopter type of flows does not allow meaningful correction of the measured data or even appropriate extrapolation of results from flight tests. Additional problems are caused by the strong interference effects between the rotor wake and the wind-tunnel walls. Consequently, for exact simulation it is somewhat more complicated, for a helicopter model to be appropriately sized for rotor scale and Reynolds number than a fixed wing counterpart.

Rotor
The first objective of testing an isolated rotor is to obtain accurate measurements of unsteady pressure distributions, which in turn can be used for computing the local flow conditions and for estimating local loads and pitching moments. This type of data is very useful from the point of view of aerodynamic performance of the rotor and especially for validating CFD prediction methods.
In the past many interesting test experiments were devoted to the investigation of rotor aerodynamics. The experiment carried out at the University of Maryland[1] was already considered a valid reference test case.
Boeing-Vertol, US Army and NASA Ames[2] carried out a very extensive experimental program at DNW wind tunnel. These tests were conducted on a fully instrumented B360 rotor scaled down to a 1/5 size. Fourteen Kulite transducers were installed on each of the 5 sections to obtain a complete description of the pressure distributions.
Another interesting experiment was carried out by UTRC (United Technologies Research Center)[3,4] and US Army with an instrumented model of the UH-60 Black Hawk helicopter rotor. The test schedule dealt with a basic rotor as well as with a modified version having a tapered tip. The basic rotor had 176 kulites, while the modified model was equipped with 96 transducers. The tests were also performed at DNW both for hover conditions and then for forward flight. Under hover conditions measurements were also carried out in the rotor wake using laser velocimetry.
In Europe, a high quality data base of simultaneously measured acoustic and blade pressures for a hingeless helicopter main rotor was acquired in the DNW facility as part of a cooperative research programme HELINOISE[5]. This experimental study employed a 40% geometrically scaled and highly instrumented main rotor model of the BO-105 helicopter.
Another relevant test was performed in the frame of the HELISHAPE project[6]. A Modular Wind-Tunnel Model (MWM) from DLR rotor test was used to drive a rotor. The rotor was fully articulated with modern airfoil designs. The model was completely instrumented and equipped with exchangeable blade tips. A comprehensive set of concurrent acoustic and aerodynamic blade surface pressure data as well as blade dynamic and performance data was measured. In addition, valuable information on the tip-vortex geometry and blade-vortex miss distance was obtained by LLS flow visualization.

Fuselage
A certain number of experiments on isolated fuselage have been carried out both in US[7,8] and Europe. In particular within the framework of GARTEUR (Group for Aeronautical Research and Technology in Europe) DLR performed a series of tests on three model helicopter fuselages with different rear parts in a 3mx3m low speed wind tunnel in Germany[9].
The models were all instrumented with pressure taps, and forces were measured with a strain gauge balance installed inside the model. Some flow field survey was also preformed to study the effects of different upsweep of the rear part of the fuselage. Data are available and already used for validation purposes.
Other studies of the same type have been conducted in UK[10] and France at ONERA-CERT[11] where again LDV measurements in the wake of the fuselage and in the boundary layer were carried out.

Rotor-Fuselage
In the AGARD-AR-303-E7[1,12] the experimental results produced at the University of Maryland are available for validation purposes. The specific data were obtained from a study investigating the mutual aerodynamic interactions between a four-bladed fully articulated rotor and a helicopter like fuselage (body), as well as between the rotor and a fixed lifting surface (wing). The objective of the test was also to try to isolate some of the important interaction mechanism.
A very useful data base was acquired by the US Army[13] during a test program in the NASA Langley 14ftx22ft wind tunnel. Two sets of rotor were tested with rectangular blades and with tapered tip blades at several advance ratios. The measurement were made along 12 radii just above the rotor tip path plane with an azimuthal space of 30 degree. This allows a very detailed description of the inflow velocities.
A realistic scaled Dauphin (365N) powered model was tested in the ONERA S2Ch wind tunnel[14]. From this model steady and unsteady pressure data on the fuselage were acquired. Also, 3D laser velocimetry (LV) data were acquired in two vertical planes.

3.4.3 CFD Modelling

The CFD codes for rotorcraft flow field studies are generally based on incompressible formulations of Euler or Navier-Stokes equations. The incompressible numerical models are suitable for describing most of the phenomena resulting from low Mach number regimes typical of helicopter flight envelopes. However, for modern rotors and faster helicopters, one may have to take into account blade tip speeds well into transonic flow regimes. Thus more sophisticated computational flow models may be required which address the compressibility effects. It is only in recent years that more rigorous numerical models based on Euler and or Navier-Stokes methods have begun to be used extensively for simulating helicopter flow fields[15,16,17] including complete configurations. Previously, only potential flow based panel methods such as VSAERO were used for calculating surface pressure measurements and other loads and pitching moments. Validation of modern Euler and Navier Stokes codes which use moving grid methods to represent the rotor would require accurate flow field and surface pressures measurements as well as other loading and pitching moment coefficients evaluations around realistic configurations.

3.4.4 Conclusions and Recommendations

Any future experimental and/or numerical code validation exercise must attempt to address the critical issues that have been highlighted in the paper.

Most of the tests reported in literature are either complete load coefficients or in some case pressure measurements recorded at various stations of the helicopter rotors. There is little evidence of flow field measurements and/or other rotor- body interaction measurements.

It seems that a real progress in the field could be achieved if an experiment could be designed around the following features:

- using the model of a realistic configurations (Fig.3.4-1 or Existing BELL 412 or AGUSTA models),
- if one of these configurations is selected there will be existing flight and/or wind tunnel data upon which we may be able to build a more comprehensive data base for future validations,
- use of a typical standard equipment (pods, fuel tanks, emergency kits) which a basic helicopter configuration is expected to carry as mission kits,
- testing in different wind tunnels, would allow a better insight into the wall correction increments used in different tunnels,
- conducting both powered and non-powered tests to study the rotor effect,
- accurate flow field measurements would improve our understanding of various interactions between different helicopter components,
- detailed flow field, pressure, load and moment evaluation to be used for CFD code validation. Pressure sensitive paints and other advances laser velocimetry may be used for flow field and surface pressure measurement.
- an accurate uncertainty analysis of the measured data.

Other initiatives that should be mentioned are from ONERA that plans to develop a complete helicopter model in S1MA and within the frame of the European project HELIFLOW where a lot of data will be collected on the Bo105. The availability of such experiments should be checked.

Acknowledgements
The authors wish to thank Dr. P. Renzoni head of Helicopter Aerodynamics Group at CIRA and Dr. M. Costes from ONERA for their valuable contributions.

References:

1. Leishman J. G and Nai-pei Bi, "Experimental data on the Aerodynamic Interactions Between a helicopter rotor and an airframe". AGARD-AR-303 Vol. 1 and 2, Published August 1994.
2. L. Dadone, S. Dawson, D. Boxwell,D.Ekquist, "Model 360 rotor test at DNW - Review of performance and blades airload data"; 43rd AHS Forum, May 1987

3. P.F. Lorber, R.C. Stauter, A.J. Landgrebe, "A comprehensive hover test of the airloads and airflow of an extensively instrumented model helicopter", 45th AHS Forum ,1989

4. P.F. Lorber, " Aerodynamic Results of a pressure-instrumented model rotor test at the DNW." 46th AHS Forum AHS,1990

5. W. Splettstoesser, B. Junker, K.J. Schultz, W. Wagner, W. Weitemeyer, A. Protopsaltis, D. Fertis, " The HELINOISE Aeroacustic Rotor Test in the DNW - Test Documentation and Representative Results", DLR-Mitt. 93-09,1993

6. K.J. Schultz W. Splettstoesser, B. Junker, , W. Wagner, E. Schoell, E. Mercker, K. Pengel, G. Arnaud, D. Fertis, "A Parametric Wind Tunnel Test on Rotorcraft Aerodynamics and Aeroacustics (HELISHAPE) Test Procedures and representative Results", 22nd European Rotorcraft Forum, 1996

7. C. E. Freeman, R. E. Mineck, "Fuselage Surface Pressure Measurements of a helicopter Wind Tunnel Model with a 3.15- Meter Diameter Single Rotor"NACA TM 80051, March 1979

8. Chaffin, M. S and Berry, J. D.,"Some helicopter fuselage pressure distribution measurements" NASA TM 4566, 94-A-013.

9. S.R. Ahmed, J.Amtsberg, "An experimental study of the aerodynamic characteristics of three model helicopter fuselages", 13th European Rotorcraft Forum, September 1987

10. J. Seddon "Aerodynamics of the helicopter rear fuselage upsweep"

11. 8th European Rotorcraft Forum, Sept 1982

12. G. Gleyzes, X. De Saint-Victor, A. Cler " Experimental study of the flow around an helicopter fuselage. Comparison with three-dimensional boundary-layer calculation", 14th European Rotorcraft Forum, Sept 1988

13. J.G. Leishman, N.Bi, D.K.Samak,M. Green, "Investigation of aerodynamic interactions between a rotor and a fuselage in forward flight. 45th AHS Forum, May 1989

14. J.W. Elliot, S.L.Althoff, R.H.Sailey, "Inflow measurements made with a laser velocimeter on a helicopter model in forward flight. NASA TM 100541, April 1988

15. J. Berry, N. Bettschart, "Rotor-Fuselage interaction: Analysis and Validation with Experiment" 53rd AHS Forum, Virginia Beach, May 1997

16. 15. J.C. Narramore, A.G. Brand, "Navier Stokes Correlations to Fuselage Wind Tunnel Test Data" 48 th AHS, Washington, DC, 1992

17. J.C. Narramore, " Navier-Stokes Computations of Fuselage Drag Increments," 49th AHS Aeromechanics Specialists Conference, San Francisco, California, 1994.

18. M. Khalid, Y. Sun, "Numerical Simulation of Rotor-Fuselage Interaction",

19. Canadian Aeronautics and Space Journal (CASI), Sep 1997
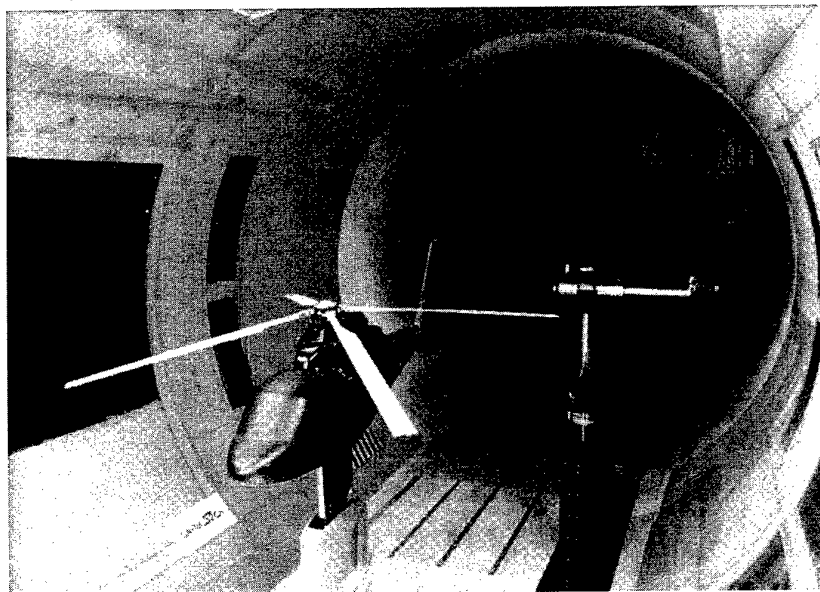
Figure 3.4-1 Dauphin Model in the ONERA S2CH wind tunnel (Courtesy of ONERA)

28

4   DISCUSSION AND CONCLUSIONS

1. The need for validation experiments for different types of configurations has been reviewed extensively:

   - For fighter-type configuration there appears to be a need for new CFD validation tests, provided that a modern design with a stealth-type signature is studied. Detailed flow field studies are required for mixed attached and separated flow conditions in combination with shock waves and vortices. Since a representative 'generic' model is not available, a new generic stealth-type configuration and the corresponding windtunnel model has to be designed and manufactured.

   - Various programs in Europe (GARTEUR, programs of the European Commission) and in the USA (NASA sponsored programs) are already concerned with transport type configurations. There appears to be a need for more detailed validation experiments in the low speed / high lift regime concentrating on viscous flow effects, involving detailed flow field and boundary layer measurements. Already existing windtunnel models that have been tested in the past can possibly be used like the DLR 'ALVAST' model and the B737-100 model used in the TSTV program. An added value for a multi-facility experiment could be obtained by concentrating on flow field measurements that require considerable expertise.

   - For missile configurations a rather large data base for validation purposes is available. However for complex flow situations, involving various control surface combinations and lateral jets, more validation experiments are required. The need to perform these experiments in different facilities is less clear however.

   - During the study helicopters have been added in response to suggestions made in the course of the study. CFD is only starting to be used in the design process of helicopters in view of the difficulty to model the very complex flow phenomena and there is a clear need for multi-facility testing to combine expertise from different groups. Already available windtunnel models like the Dauphin, the Bell 412 or some Agusta configurations are candidate windtunnel models for such a validation experiment.

2. It was also found from responses to a questionnaire distributed by the Working Group within the CFD community concerning the state of the art for CFD validation that half of the respondents found the situation acceptable with satisfactory and disappointing experiences equally divided on both sides. Most of the respondents indicated a need for more detailed measurements. It was reported that there was a lack of validation experiments for helicopters.

3. For all cases complex flow phenomena, involving interactions of different flow types are an essential part of new validation experiments; in this respect it is equally essential to apply newly developed, challenging measuring techniques.

4. The Working Group was established to find out if there is a need for comparative, multi-facility windtunnel testing for CFD validation to eliminate tunnel specific bias effects. It is found that CFD validation experiments to-day require detailed flow field measurements for complex, representative windtunnel models specifically designed for validation purposes. It is argued that the technical need and cost effectiveness of multi-facility testing to reduce possible bias effects in relation with CFD validation, is rather limited. This was a shared feeling within the Working Group and inquiries made within the different organisations that were represented in the Working Group, supported this view.

5. However it is still of interest to establish bias errors between various facilities using comparative tests with one or more simple standard models, specifically designed and instrumented to understand the cause of possible differences. Such an exercise should be viewed in the context of 'quality assurance'.

6. Collaborative, multi-facility testing will widen the scope and the technical depth of the experiment. It will be cost effective to use the experience of different groups of experts for advanced measuring techniques in a complementary manner.

7. The Working Group members have made a 'bottom-up' inquiry within their own organisations to find out if one or more of the programs mentioned above could find support. In all cases the response was inconclusive. Cost issues and commercial / political reasons were quoted to limit the chances for a multi-facility exercise. It is believed that such a program could only be envisaged when there is a clear top-down commitment from the organisations involved.

REPORT DOCUMENTATION PAGE

1. Recipient's Reference	2. Originator's References	3. Further Reference	4. Security Classification of Document
	RTO-TR-27 AC/323(AVT)TP/21	ISBN 92-837-1023-1	UNCLASSIFIED/ UNLIMITED

5. Originator
Research and Technology Organization
North Atlantic Treaty Organization
BP 25, 7 rue Ancelle, F-92201 Neuilly-sur-Seine Cedex, France

6. Title
A Feasibility Study of Collaborative Multi-facility Windtunnel Testing for CFD Validation

7. Presented at/sponsored by
the RTO Applied Vehicle Technology Panel (AVT) Working Group 011.

8. Author(s)/Editor(s)	9. Date
Multiple	December 1999

10. Author's/Editor's Address	11. Pages
Multiple	40

12. Distribution Statement There are no restrictions on the distribution of this document.
Information about the availability of this and other RTO
unclassified publications is given on the back cover.

13. Keywords/Descriptors

Wind tunnel tests	Fighter aircraft
Wind tunnels	Transport aircraft
Computational fluid dynamics	Rotary wing aircraft
International cooperation	Helicopters
Algorithms	Missiles
Experimental data	Flow distribution
Computation	Flow measurement
Comparison	Test facilities
Validity	Cost effectiveness

14. Abstract

Between 1997 and 1999, a Working Group on the feasibility of collaborative, multi-facility windtunnel testing for the validation of Computational Fluid Dynamics Codes (CFD) has established the following results:

Windtunnel tests are the main source of CFD validation but all have specific bias errors. Testing of the same model in several facilities can better establish the level of uncertainty in windtunnel tests and hence help assess possible differences between CFD and windtunnel tests.

The working group, after an initial inquiry, established the need for CFD validation for fighter aircraft, transport type aircraft, missiles and helicopters. Effort and cost can be controlled efficiently and would permit the improvement of CFD Codes for considerable cost saving in future developments.